A NUMERICAL SIMULATION OF THE EFFECTS OF POSITION AND SPRAY ANGLE OF THE FUEL INJECTION NOZZLE ON MIXING IN A CIRCULAR COMBUSTOR_

Bahman Ghorashi and Sastry Taruvai

Chemical Engineering Department

Cleveland State University

Cleveland, Ohio 44115

ABSTRACT

A Numerical study was conducted using Fluent computer program developed by Creare, Inc. The objective was to determine the flow pattern between two sets of baffles in an experimental reverse-flow circular combustor, designed for enhanced mixing and NO_x reduction, by varying the position and spray angle of the fuel injection nozzles. The segment of the combustor that was investigated consisted of two baffles positioned on the upperface and two on the lowerface of the circular combustor. Heated air at 300°F and 3 atmospheres pressure was introduced into the combustor with an inlet velocity of 100 ft/sec. The fuel, pentane, was introduced into the main stream via fuel injection nozzles. There were four nozzles, two on the upper and two on the
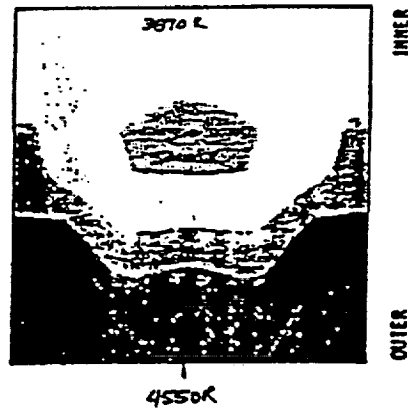
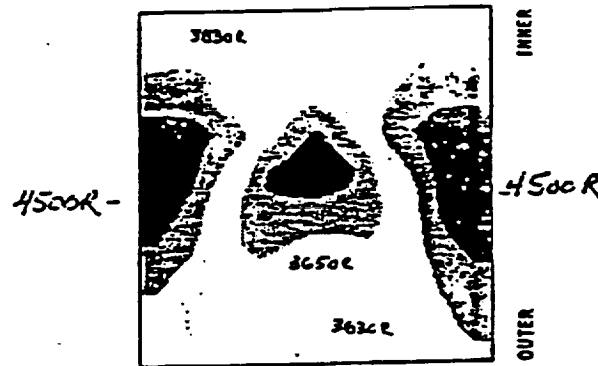Figure 15 – Exit Section Temperature Distribution for Case 6

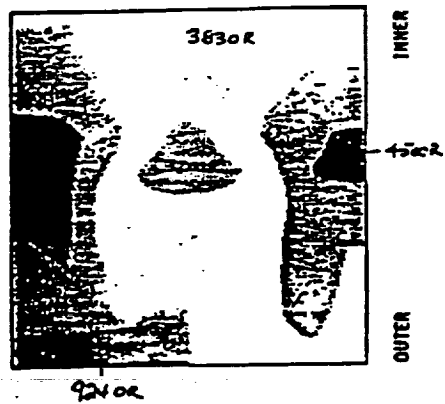Figure 13 - Exit Section Temperature Distribution for Case 4

Figure 14 - Exit Section Temperature Distribution for Case 5

MIDWAY

EXIT

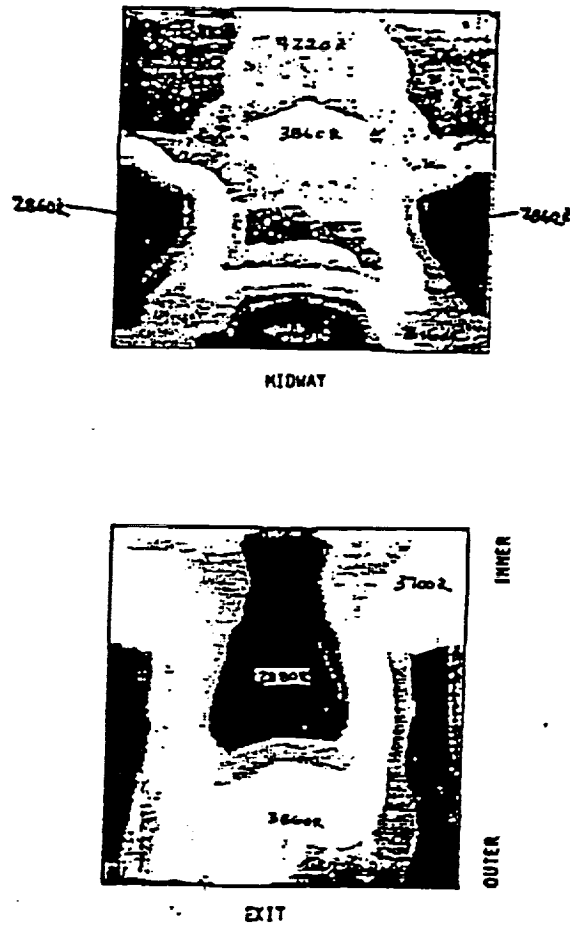Figure 12 – Cross-Sectional Temperature Distributions for Case 3

MIDWAY

EXIT
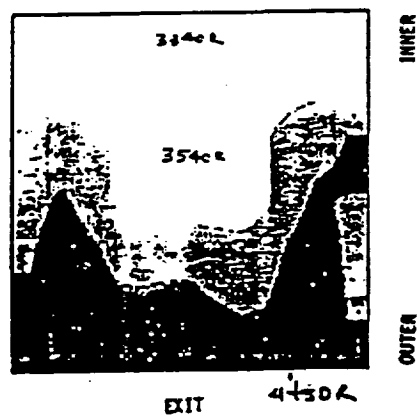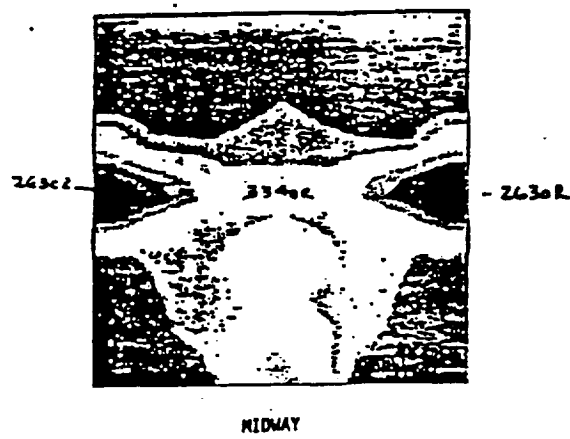
Figure 11 - Cross-Sectional Temperature Distributions for Case 2

SIDEWALL   CENTERLINE
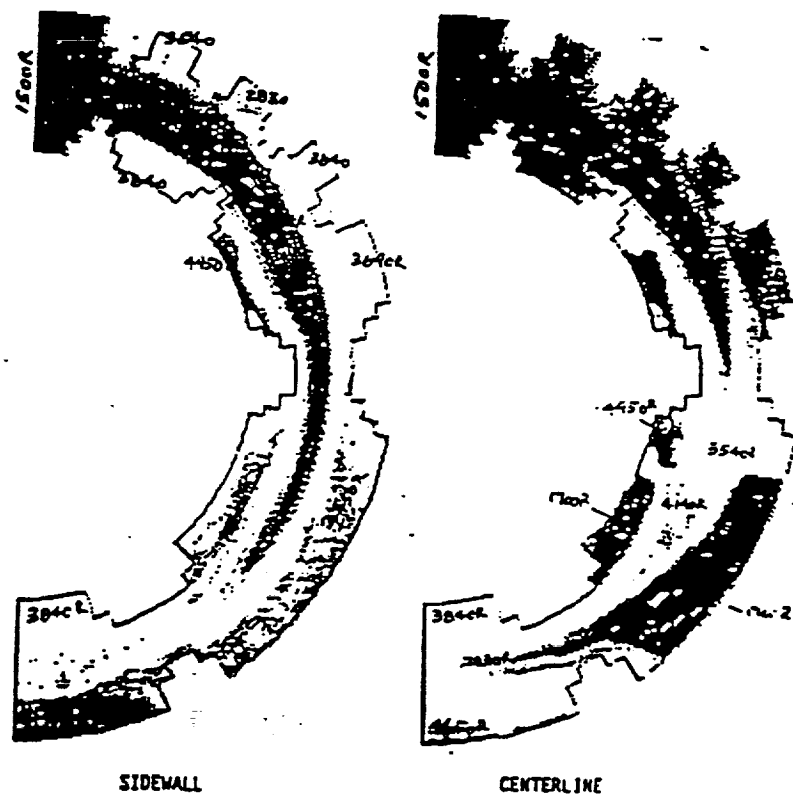
Figure 10 – Temperature Distributions in Combustor for Case 2

MIDWAY

EXIT
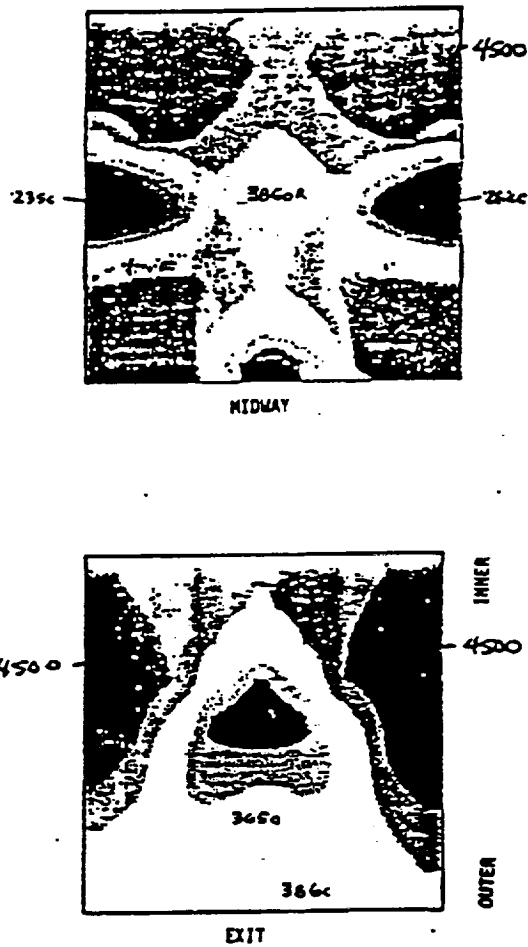
Figure 9 – Cross-Sectional Temperature Distributions for Case 1

STDEWALL   CENTERLINE
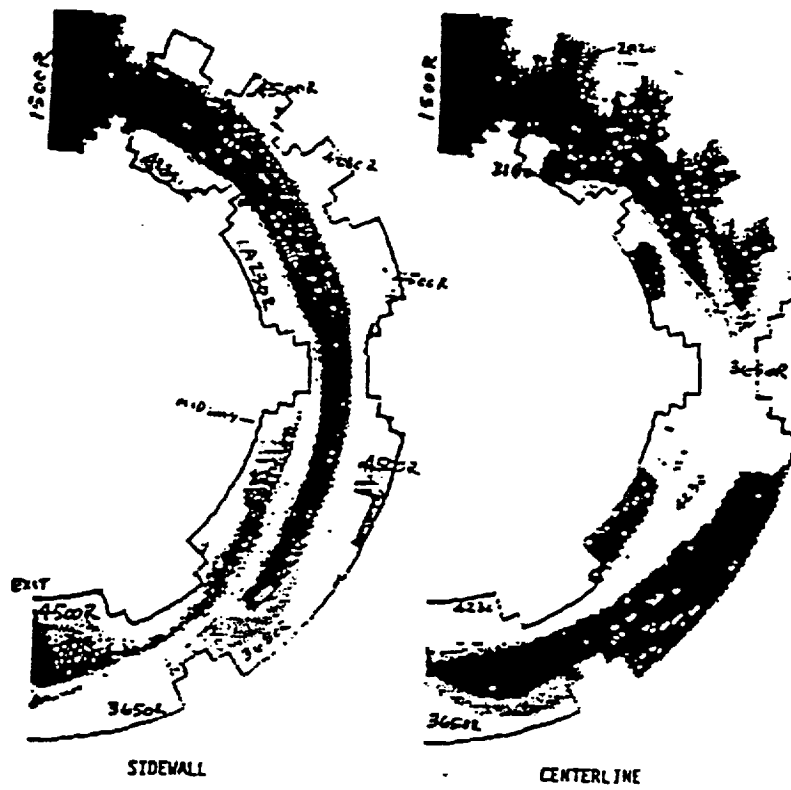
Figure 8 - Temperature Distributions in Combustor for Case 1
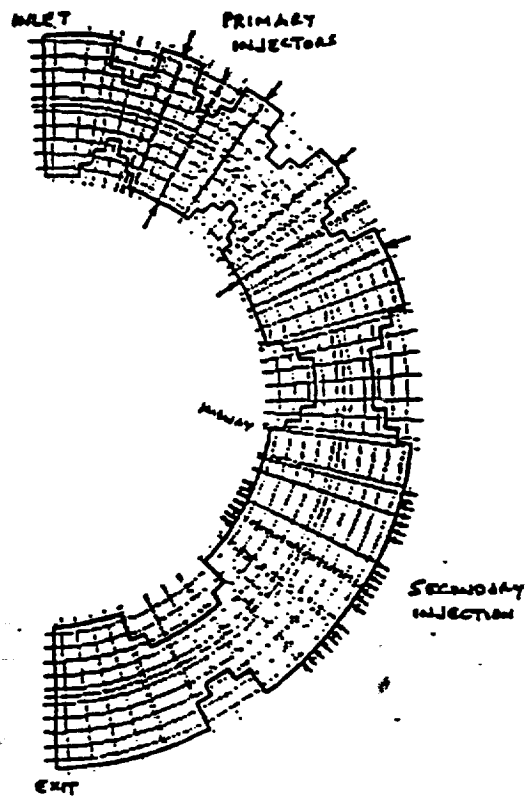
Figure 7 – Finite Difference Grid for Combustor Model
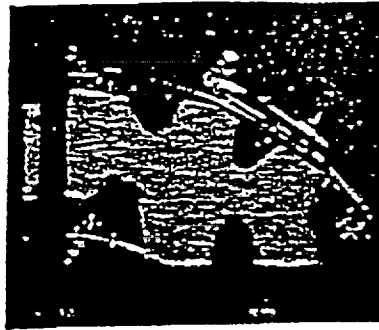
Figure 5 -A Single Frame of the Flow Pattern in the Reverse Flow Combustor.
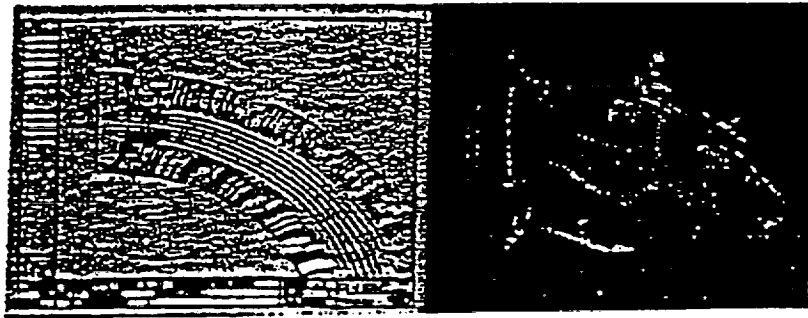
Figure 6 -   A Comparison between the Numerical Simulation of the Flow and a Randomly Selected Flow Visualization Image.
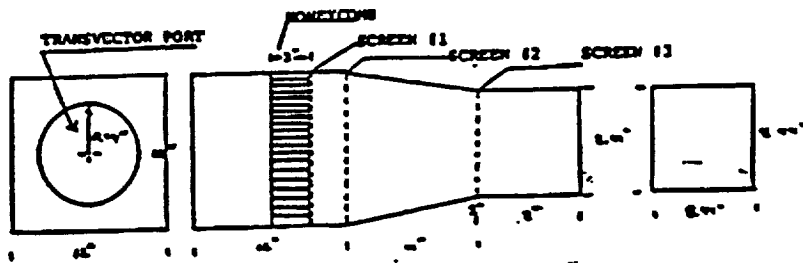
Figure 3 – A Schematic Diagram of Flow Straightner Unit.
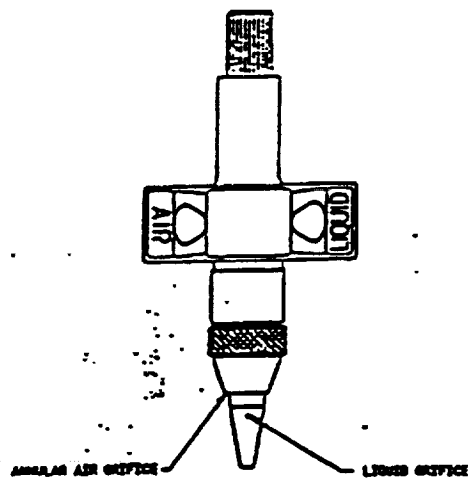
Figure 4 – Typical Air-Blast Fuel Injector
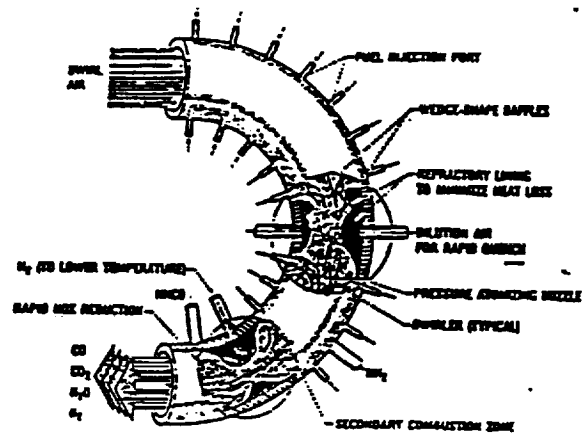
Figure 1 - Schematic Drawing of the Reverse Flow Combustor.
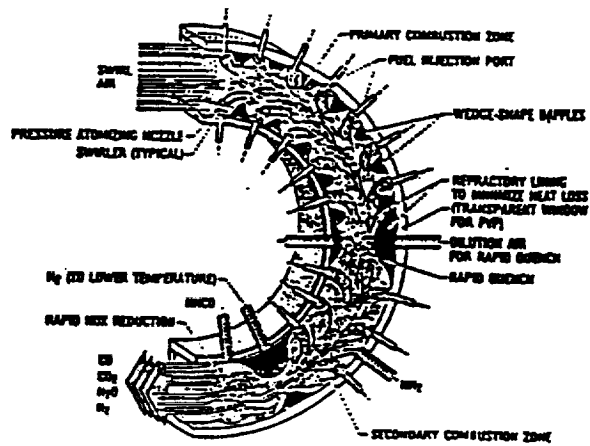
Figure 2

Schematic Drawing of the Region of Interest in the Reverse Flow Combustor.

REFERENCES

1.   Ghorashi, B., Et al., "A Circular Combustor Configuration Design With Multiple Injection Ports for Mixing Enhancement," Proceedings of the 31st Heat Transfer and Fluid Mechanics Institute, California State University, Sacramento, 1989.

2.   Ghorashi, B. and Taruvai, S., "A Numerical Simulation of the Effect of Position and Spray Angle of the Fuel-Injection Nozzle on Mixing in a Circular Combustor", Presented at the 6th Miami International Symposium on Heat and Mass Transfer, December 1990.

3.   FLUENT User's Manual, Creare-X, Inc., Hanover, New Hampshire, 1988.

4.   Reardon, F. H., and Clifford, R. J., "Calculation of the Combustion Distribution in a Liquid-Fuel Ramjet," 26th JANNAF Combustion Meeting, Jet Propulsion Laboratory, Pasadena, CA, 1989.

combustor exit.

Figure 9, compares the temperature distributions for two cross sections for Case 1. The section labeled "Midway" is just downstream of the mid-combustor baffles. The cool flow regions near the side walls can be seen clearly. Some effect of the secondary air can be seen near the outer wall. At the combustor exit, the coolest region is at the center of the cross section, with the hottest zones near the side and inner walls.

Similar temperature pattern can be seen in Figures 10 and 11 for Case 2. The primary zone for this case is quite similar to that of Case 1. However, near the combustor exit there is a significant difference. Although the coolest region is the center for both cases, the hottest region is near the outer wall in Case 2 and nearer the inner wall for Case 1. Since the only difference between the two cases is the fuel injection velocity, the reason for this difference is not clear.

Figure 12, shows the cross section temperature distributions for Case 3, which has a smaller injection spray angle than Case 1. The distributions are similar for the two cases, except that at the "Midway" section Case 3 shows the cool regions nearer the outer wall and at the exit the hottest regions are nearer the outer wall.

The effect of fuel drop size on the temperature distribution at the combustor exit can be seen by comparing Figures 9, 13 and 14. In all three cases, the hottest regions are near the side walls, with a cooler center core region. The cause of the asymmetry in Figure 14, is not known at this time.

The exit cross-section temperature distribution for Case 6, Figure 15 is most similar to that for Case 2, even though the latter case has a single line of injectors along the centerline of the combustor and the spray angle is 45° whereas the Case 6 there are five lines of injectors and a zero spray angle. The fuel injection velocity is the same for both cases. Both cases show higher temperature regions across the entire outer combustor wall.

Further studies are needed to gain insight into the differences between the temperature distribution patterns.

CONCLUSION

The pattern depicted by the smoke visualization technique showed a behavior which is very desirable for the rapid mixing of fuel and air. These studies indicate the need for more effective atomization of the fuel. Low injection velocity and large injection spray angle were found to produce a more uniform temperature distribution at the exit. Ideally, the fuel injector nozzle should be capable of controlling swirls fuel spray angle, and jet penetration.

TABLE II
SUMMARY OF COMBUSTOR CASES
AND EXIT TEMPERATURE RESULTS

Case Number	Injection Velocity ft/sec	Spray Angle Degree	Drop Diameter Micron	Number of Injection Streams	TMIN Degree R	TMAX Degree R	Location of TMAX (Wall)	TMEAN	TMAX TMIN
1	120	45	50	24	2970	4630	Side	3980	0.417
2	30	45	50	24	3480	4390	Outer	4100	0.222
3	120	10	50	24	2700	4530	Side	3740	0.489
4	120	45	25	24	2900	4600	Side	3930	0.433
5	120	45	100	24	3330	4630	Side	4020	0.323
6	30	0	50	30	3220	4670	Outer	4110	0.353

The mean temperature at the combustor outlet varied from 3742 °R for Case 3 (small fuel spray angle) to 4108 °R for Case 6 (30 fuel injectors). Case 2 (low fuel injection velocity) gave a mean temperature nearly as high as Case 6 as well as the most uniform temperature distribution. It is interesting that the least uniform distribution was found for Case 3, which had the lowest value of the mean temperature. Since the fuel drop size was the same for Cases 2 and 3, the difference in the results can be attributed to the initial fuel dispersion in the air stream.

Comparing Cases 1 and 2 shows that the greater fuel penetration into the combustor volume associated with the higher fuel injection velocity had a detrimental effect on the temperature distribution. The narrower angle of the fuel spray of Case 3 relative to that of Case 1 resulted in additional nonuniformity of the temperature at the combustor exit. Perhaps the zero spray angle used for Case 6 is the reason that it did not show as uniform a temperature distribution as Case 2, even though the fuel sprays were more uniformly distributed.

A more interesting result was that a larger drop diameter gave a higher mean temperature as well as a more uniform distribution, as can be seen by comparing Cases 1, 4, and 5. The longer burning times of the larger drops may have led to more uniform mixing with the air.

It must be kept in mind that the primary combustion zone, where the fuel sprays are located, is fuel-rich, with a fuel-air equivalence ratio that increases with distance from the inlet up to a value of 1.40. Air introduced in the secondary zone has a strong effect on the temperature distribution. Figure 4, shows the temperature distribution for Case 1 along the length of the combustor, at the centerline and near the side wall. Along the centerline, moderately high temperatures are developed by the end of the primary zone. The dilution air introduces a cool core flow that persists to the combustor exit. However, at the side wall there is a cool core flow well into the secondary combustion zone. The highest temperatures are found at the sides near the inner wall at the

Case 2:

The fuel injection velocity was set at 20 ft/sec for this case. Such a velocity could be achieved by increasing the fuel hole diameter and/or the number of holes. Even though such a reduction in velocity would be likely to decrease the fuel spray angle, for this case only the velocity was reduced. Al other parameters were unchanged from the baseline case.

Case 3:

The atomizer manufacturer's data indicated that the typical spray would have a spreading angle of only about 10 degrees, rather than 45. Therefore, the angle in this case was set at 10 degrees. However, the fuel velocity was restored to the baseline value of 120 ft/sec.

Case 4:

The effect of mean fuel drop size was checked by keeping all input parameters the same as the baseline, but decreasing the fuel droplet diameter to 25 microns. This was about the size resulting from the theoretical calculations made on the injector/atomizer part of the project.

Case 5:

For this case, the fuel drop size was increased to 100 microns, which is closer to the mean drop size given in the available data. All other input values were available the same as for the baseline case.

Case 6:

In this case, the single line of injectors along the combustor center line was replaced by five lines of injectors spread across the width of the combustor. Such a design would allow the total fuel flow to be supplied with a lower injection velocity than for the baseline case. The fuel would also be more uniformly distributed. To simplify the input, the fuel spray angle was assumed to be zero; that is a single stream of fuel drops was injected along the centerline of each fuel injector.

RESULTS OF CALCULATIONS

The results of the FLUENT calculations are summarized in Table II and presented graphically in Figures 8 to 15. Since the flow in the combustor is quite complex, simple cause-and-effect relationships are not to be expected.

midsection of the combustor, the other downstream of the secondary air injection nozzles. These baffles are larger than the primary combustor baffles.

Figure 7, shows the finite difference grid used. The basic grid is 13 x 13 x 76, with additional grid lines inserted at each fuel injector location. The injectors are modeled as a radical-flowing air jet and a fuel droplet array. The air jet is sized so that its cross-sectional area and velocity give the proper air flow for fuel atomization (based on a Sprayvector atomizer with air injection through an annulus 0.5 inch in diameter and 0.002 inch high). The fuel spray is modeled as four streams of droplets corresponding to the four liquid injection holes in the Sprayvector atomizer. All fuel streams start from locations in the combustor corresponding to the ends of the atomizers, about one inch from the combustor wall. The droplet diameter, velocity, and the direction of the droplet motion (measured with respect to the centerline of the injector, which is normal to the combustor wall) are inputs to the calculation that can be varied for each case calculated. The fuel mass flow rate was adjusted to produce an overall equivalence ratio of 1.4 in the primary combustion zone.

Table I shows the combustor inlet conditions as specified for the computational studies. Because Version 2.99 of FLUENT cannot handle Mach numbers above about 0.3, the inlet velocity was reduced to 100 ft/sec. The secondary air is introduced at 210 ft/sec, with the flow area adjusted to produce an overall equivalence ratio of 0.7.

TABLE I COMBUSTOR INLET CONDITIONS

Operating Conditions	Pressure	Temperature	Velocity
Idle	3 atm (44.1 psia)	422 K (760 R)	75 ft/sec
Cruise	14 atm (205.8 psia)	833 K (1500 R)	500 ft/sec
Cold Flow	1 atm (14.7 psia)	298 K (536 R)	25 ft/sec

Case 1 – Baseline:

For this case, the fuel injection velocity was 120 ft/sec. This value was obtained from the fuel flow rate required, and assuming that each injector had four fuel orifices, 0.047 inch in diameter. Because of the high fuel velocity, it was estimated that the fuel spray angle with respect to the injector centerline was 45 degrees. The drop diameter was assumed to be 50 microns.

To aid the flow visualization of the air blast fuel injector, yellow dyed smoke was injected via a high pressure vessel in the air line feeding the fuel injector. The pattern was photographed using color film. The pattern was that of typical expanding turbulent jet when there was no main flow and bent concurrently in the direction of flow as the main air flow was introduced.

High speed films were studied to identify different flow patters in the MCZs, using a 16 mm film projector with speed control capability. After viewing 16,000 frames per film, a frame by frame advance was conducted in predetermined segments of each film to examine the flow pattern in greater detail. In Figure 5, for example, a single frame of the flow pattern in the reverse flow combustor is depicted at 1/100th second, 5/100th second, and 10/100th second intervals. The first frame shows the very high velocity parabolic jet profile of the smoke entering the first MCZ. As the jet enters the first pair of MCZs, it encounters the first set of air blast fuel injectors perpendicular to the flow and begins to experience the effects of the second pair of baffles. This phenomenon is enhanced due to the curved wall of the combustor positioning the baffles relative to the main flow. A portion of the flow is seen recirculating into the MCZ in the second and third frames.

Figure 6, illustrates a comparison between the numerical simulation of the flow and a randomly selected flow visualization image from the same spatial position. Strong similarities are shown between the simulation, by using a computational fluid dynamics code [2] and the flow image. Both images show a high velocity main channel flow accompanied by MCZ recirculation.

A series of experimental runs were completed to study liquid and main flow gas interaction. The main flow was set to 2500 SCFM (at 100 psig). The air blast injector was operated over an estimated droplet range of 60 to 170 microns based on manufacturer's data. The pressure was set at the lowest range but outside of the recommended operating pressure range (> 80 psig). The pressure range was 15 psig from the 60 micron size droplet and 25 psig for the 170 micron droplet to simulate turbine engine operating conditions. The camera was operated at 1000 frames per second to photograph very rapid air motion and its interaction with liquid droplets. The low nozzle pressure range that was used caused droplet coalescence.

COMPUTATIONAL STUDIES

Combustor Model

As noted earlier, in the primary combustion zone, there are four baffles on the outer wall and two on the inner wall These baffles are 2.8 inches which and 3.0 inches long and extend the full width of the combustor. A fuel injector/atomizer is located downstream of each baffle, midway between the two side walls. (Figure 7). The fuel injectors/atomizers are of the air-blast type, with a high velocity jet of air atomizing the liquid fuel and transporting it into the combustion volume in a direction approximately normal to the mean flow in the combustor. Two more pairs of baffles are located downstream, one at the midsection of the combustor, the other downstream, one at the

The main channel flow was generated by a sonic air amplifier. A compressed air annulus at M = 1 eductively drew room air to generate a substantial volumetric air flow. A regulator was used to set the air pressure, thereby controlling the volumetric air flow.

The air blast fuel injectors were driven by a separate, regulated air supply, (Figure 5). The 20 injectors were supplied from a custom manifold. The fuel port of the injectors can be supplied with air, smoke, or water through another manifold depending upon the choice of flow visualization experiment.

Procedure

For the purpose of flow visualization studies, experiments were completed to determine the appropriate concentration of smoke in the reverse flow circular combustor. A smoke generator with a 4500 cfm capacity was set at 1/10th its full range capacity and smoke was injected concurrently with the main stream air flow. The inlet pressure to the main air flow amplifier was 30 psig which was equal to 1000 scfm of ducted air flow in the combustor.

Smoke rapidly filled the primary combustion chamber, thereby eliminating the visual access to details in the flow pattern. This, however, did not create any appreciable difficulty, as only the initial 2 to 3 seconds of high speed camera film, at 100 frames per second, were of interest.

To maximize the recovery of photographic detail, a black non-reflective background was used behind the model with white smoke for flow visualization. Initial experiments used two 1000 watt quartz lamps. The lamps were situated above and below the primary combustion chamber to create a high intensity plane of light for the smoke to pass through. Although adequate for very close and small area photography, the total wattage was increased to three lamps at 100 frames per second and four lamps at 1000 frames per second to enhance the visual contrast of the flow.

The camera was positioned perpendicular to the flow and four feet from the combustor on the same horizontal plane. Sixteen (16) millimeter film, 400 ASA with a 25 mm lens with a 360 degree/2.5 shutter was used throughout the study. The f-stop was varied from 2.8, 4.0, to 5.6 in a discrete mode.

Results

To determine if the entire combustor volume was being utilized, flow visualization studies were conducted with blue dyed smoke injected into the main stream air flow creating a light blue background and the white smoke from the generator was injected as a concentrated, thin stream about 5/8 inch in diameter along the unbaffled wall of the combustor closest to the camera. The flow pattern was photographed with color film. The white jet stream was drawn into the main flow between the 2nd and 3rd MCZ and mixed with the dominate blue stream before entering the secondary combustor, thus no short circuiting was occurring.

four baffles on the outermost circular surface and two baffles on the innermost surface (Figure 2).

The flow pattern was studied using air at 72°F with an average velocity of 38 ft/sec. Smoke was introduced into the main flow via a smoke generator, a flow straightener, and a "reverse trombone" injection nozzle. The flow pattern was photographed with a high speed camera at 100 frames per second.

Computational fluid mechanics studies using the FLUENT computer code were undertaken to support the development of the combustor design. An overall combustor model was developed and used to determine the effects of fuel injector design on the temperature distribution throughout the combustor, with particular attention to the combustor exit.

The primary computational tool used in this investigation was FLUENT Version 2.99 [2], a general-purpose computer program for modeling fluid flow. FLUENT uses a finite difference numerical procedure to solve the Navier-Stokes equations, together with equations for turbulence and chemical species. The fluid is regarded as a continuum and the governing equations are solved in an Eulerian frame of reference. A Lagrangian approach is used for liquid droplets or particles.

Two aspects of the combustor flow were studied. An overall combustor model was developed. This model included the circular combustor shape with the internal baffles and multiple injection locations, but with a simplified representation of fuel injection sprays. This model was used to determine the effects on the temperature distribution in the combustor of some of the key fuel injection parameters, including drop size, velocity, and spray divergence angle. A separate study examined the nature of the flow from a typical air-blast injector element in the absence of cross flow and chemical reaction.

FLOW VISUALIZATION STUDIES

Equipment

A smoke generator was used to inject smoke into the main air flow. The smoke was a high molecular weight, polyhydric alcohol. The flow straightener was constructed from a 1 foot I.D. cube tapering down to an 8.5 inch I.D. cube. It contained a honeycomb screen followed by three wire screens, (Figure 3). An injection tube was constructed to control the smoke volume and the location of the smoke injection along the face of the honeycomb.

The reverse flow combustor flow visualization unit was constructed from polymethyl methacrylate (PMMA) in the geometry of an 8.5 inch I.D. duct with a 42 3/4 inch inner radius and a 62 3/4 inch outer radius. Its first-stage contains six MCZ's and six injectors and the second-stage contains 4 mixing jets. This stage does not contain MCZ's.

7N -34- TM
45501
p. 21

305

FLOW VISUALIZATION AND COMPUTATIONAL STUDIES
OF A REVERSE FLOW CIRCULAR COMBUSTOR

B. Ghorashi, F. Reardon*, G. McBeath, and K. Chun⁻
Chemical Engineering Department
Cleveland State University
Cleveland, Ohio 44115

ABSTRACT

Computational Fluid Mechanics Studies using the Fluent computer program and flow visualization studies were undertaken to support the development of a reverse-flow circular combustor design. The reverse-flow circular combustor combines a circular geometry with multiple fuel injection nozzles in Mini-Combustion Zones (MCZ). An overall combustor model was developed and used to determine the effects of fuel injector design on the temperature distribution throughout the combustor, with particular attention to the combustor exit. Low injection velocity and large injection spray angle were found to produce a more uniform temperature distribution at the exit.

The flow pattern depicted by the smoke visualization technique showed a behavior which is very desirable for the rapid mixing of fuel and air. The atomization of fuel was visualized by injection of a yellow dye smoke in the liquid side of the air-blast injector. The portion of the combustor volume that was investigated consisted of four baffles on the outermost circular surface and two baffles on the innermost surface. The flow exhibited a rapid circulation pattern inside the MCZs. The channel flow also exhibited a high degree of interaction with the MCZs. Experimental investigation of the flow pattern in the combustor showed similarities to the computational fluid dynamic results.

INTRODUCTION

The development of efficient air-breathing jet propulsion engines that produce low levels of pollutants in the exhaust will require an improved understanding of the flow and reaction processes that take place in combustion chambers. These processes include atomization and vaporization, when liquid fuels are used, and transport, mixing, and chemical reaction, for both liquid and gaseous fuels. Computational fluid mechanics is a technique for studying these processes without the need for excessive experimental runs.

A flow visualization study of the mixing pattern in a reverse flow combustor was conducted (Figure 1). A transparent model was constructed to examine the flow pattern and mixing of side-mounted turbulent jets and their interaction with the main channel flow. The portion of the combustor volume that was investigated consisted of

* Mechanical Engineering Department, California State University, Sacramento
⁻ NASA, Lewis Research Center